ADS Fundamentals - 2001

LAB 3: DC Simulations and Circuit Modeling

Overview - This chapter introduces the use of behavioral models to create a system such as a receiver. This lab will be the first step in the design process where the system level behavioral models are simulated to approximate the desired performance. By setting the desired specifications in the system components, you can later replace them with individual circuits and compare the results to the behavioral models.

OBJECTIVES

? ***Model a generic BJT with parasitics and save it as a sub circuit.***

? ***Set up and run numerous DC simulations to determine performance.***

? ***Calculate bias resistor values in the data display.***

? ***Build a biased network based on the DC simulations.***

? ***Test the biased network.***

Agilent Technologies

Table of Contents

PROCEDURE

The circuit you bulid for this lab exercise will be used as the lower level hierarchial circuit for all of the amplifier labs to follow.

1. Set up and new project and schematic.

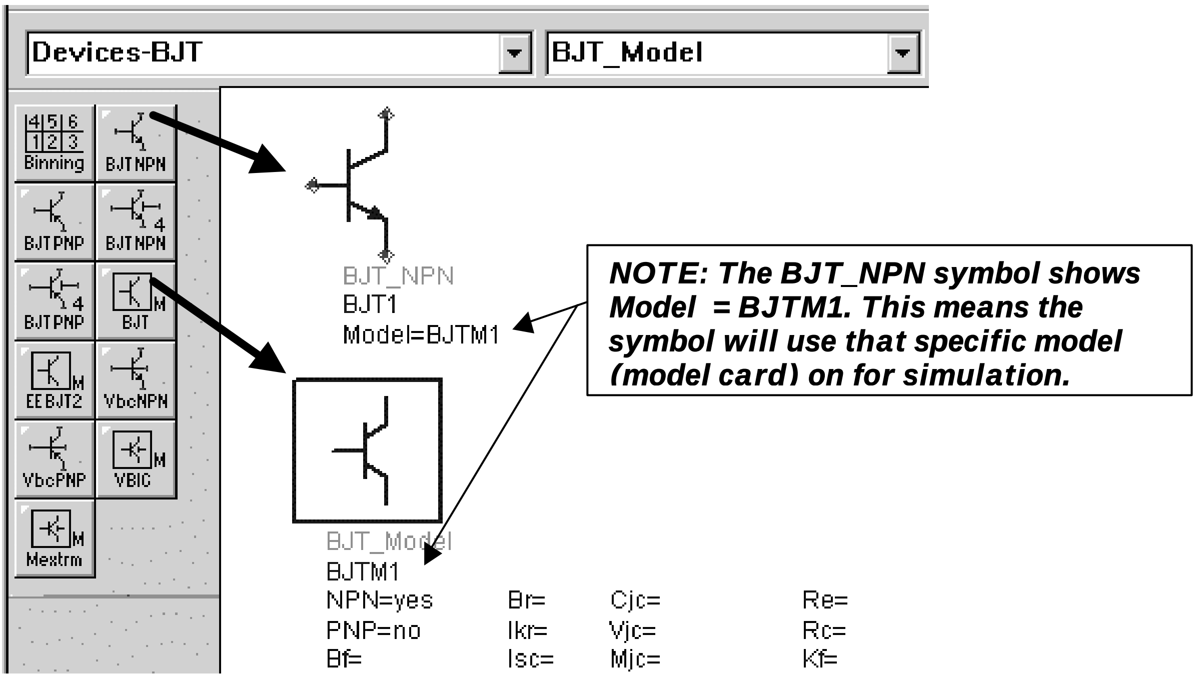
a. Create a New Project and name it: amp_1900.

b. Open a New Schematic Window and save it as: bjt_pkg

2. Set up a generic BJT device symbol and model card

a. <u>Insert the BJT generic device and model card</u>: In the schematic window, select the palette: Devices–BJT. Select the BJT-NPN device shown here and insert it onto the schematic.

b. Insert the BJT_Model (model card) shown here.

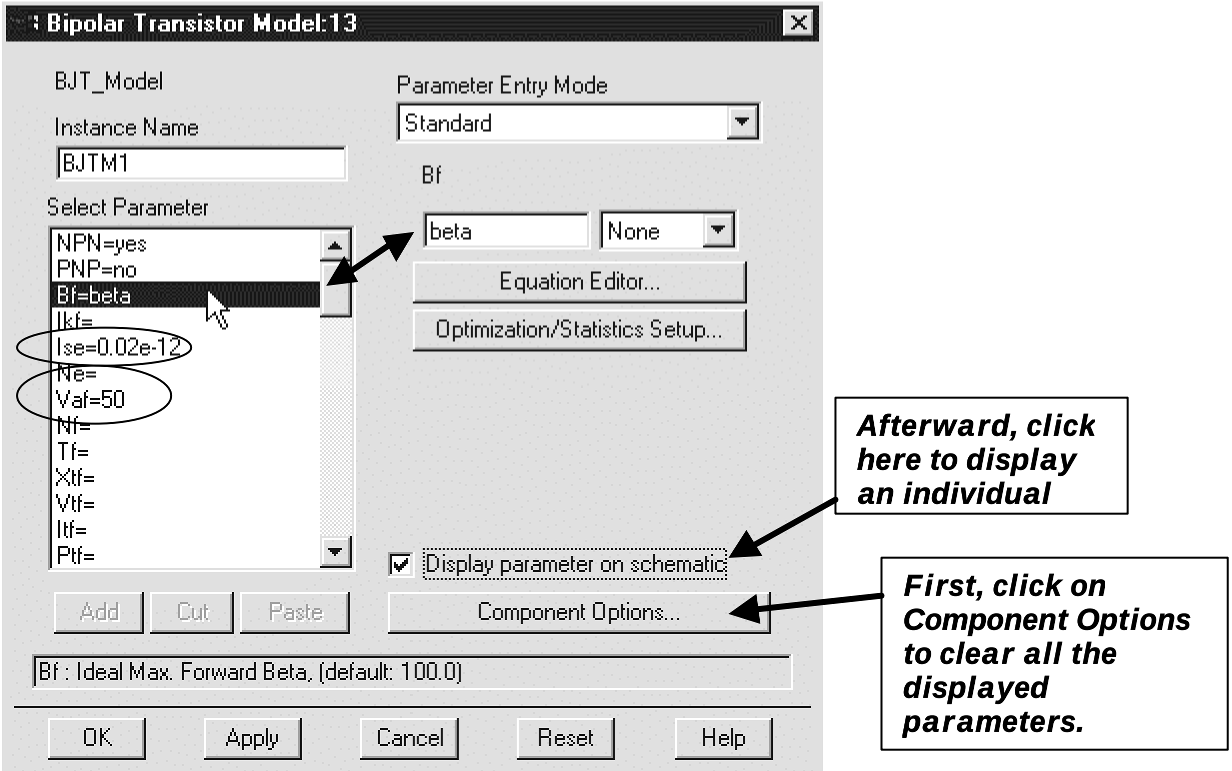


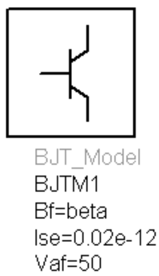
c. Double click on the BJT_Model card. When the dialog appears, click Component Options and in the next dialog, click Clear All and OK. This will remove the Gummel Poon parameter list from the schematic.

NOTE on Binning: You can insert multiple model cards and use the Binning component to vary the model device variations. These variations can be

parameters such as temperature, length or width. The Binning component allows you to create a matrix that references the desired models.

a. Double click on the BJT_Model you just inserted. Select the Bf parameter and type in the word beta as shown here. Also, click the small box: Display parameter on schematic for Bf only and then click Apply. Beta is now a variable you can tune.

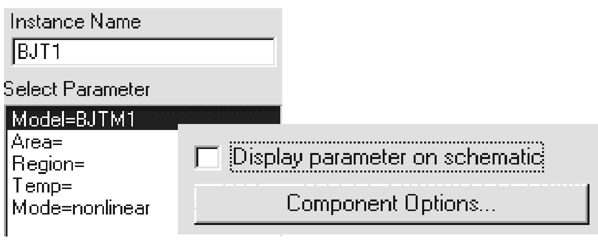


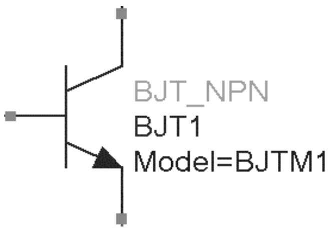


b. Set Ise (E-B leakage) = 0.02e-12, and display it by clicking the check box and Apply.

c. Set Vaf (Forward Early Voltage) = 50 and display it. These parameters give some reality to the model.

d. Finally, for the BJT device, remove the unwanted display parameters (Area, Region, Temp and Mode) by unchecking the box. This will make the schematic less crowded with parameters that you will not be using.



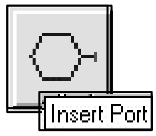


3. Add parasitics and connectors to the device.

The picture here shows the completed sub-circuit with connectors and parasitics. Remember to use the rotate icon for orientation of the components as you insert them. Here are the steps:

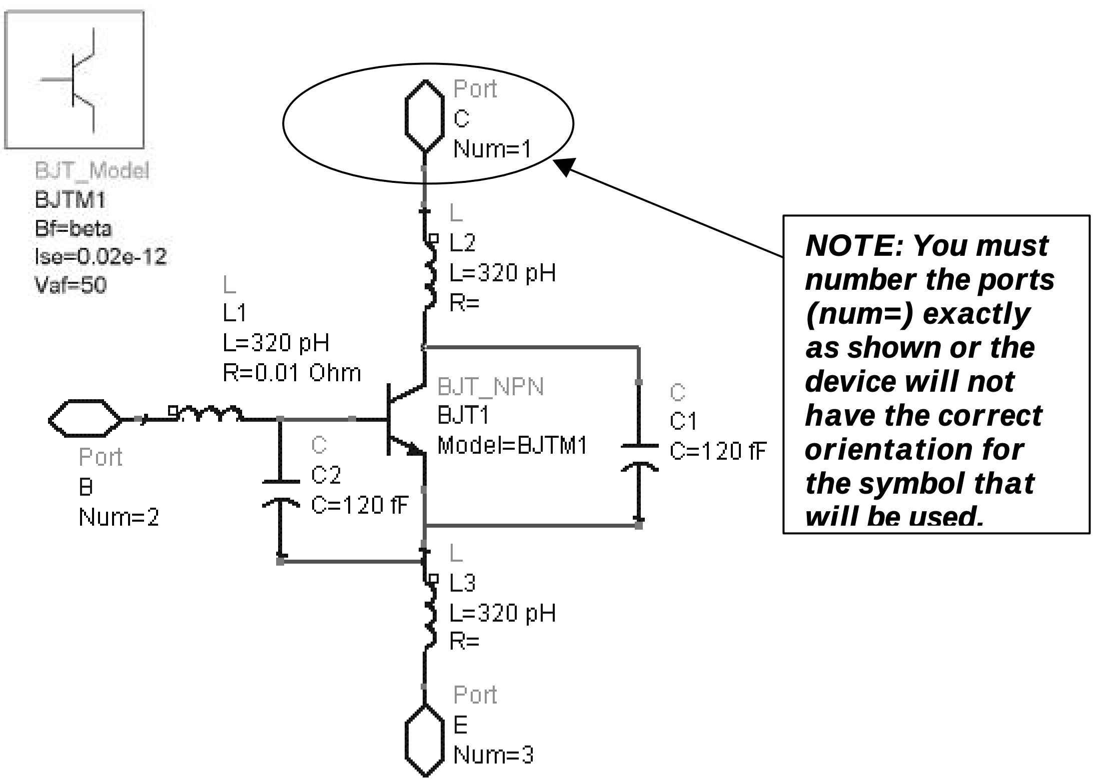
a. <u>Insert the lumped L and C components</u>: Insert three lead inductors (320 pH) and two junction capacitors (120 fF). Be sure to use the correct units (pico and femto) or your circuit will not have the correct response. Also, add some resistance R= 0.01 ohms to the base lead inductor and display the desired component values as shown.

b. <u>Insert port connectors</u>: Click the port connector icon (shown here) and <u>insert the connectors exactly in this order</u>: 1) collector, 2) base, 3) emitter. You must do this so that the connectors have the exact same pin configuration as the ADS BJT symbol.



c. Edit the port names as show here: change P1 to C, change P2 to B, and change P3 to E.

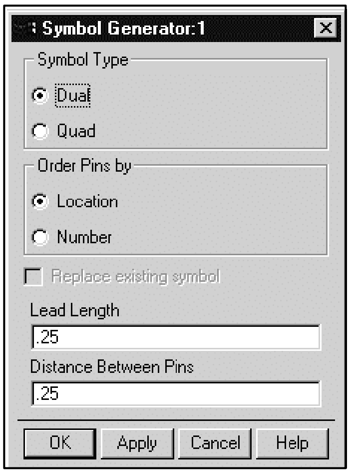
<u>Clean up the schematic</u>: Position the components so the schematic looks organized – this is good practice. To move component text, press the F5-Key and then select the component. Use the cursor to position the text.



4. Create a symbol for the sub-circuit

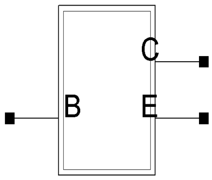
There are three ways to create a symbol for a circuit: 1) Use the default symbol, 2) Use a built-in BJT symbol, or 3) Draw your own sysmbol. For this lab you will use a built-in BJT symbol. The following steps show how to do this:

a. To see the default symbol, click: View > Create/Edit Schematic Symbol. The symbol view will replace the schematic and a dialog will appear. Click OK.



b. Next, a box or rectangle with three ports is generated. This is the default symbol. However, delete this symbol using the commands: Select > Select All. Then click the trash can icon or delete key.

Default

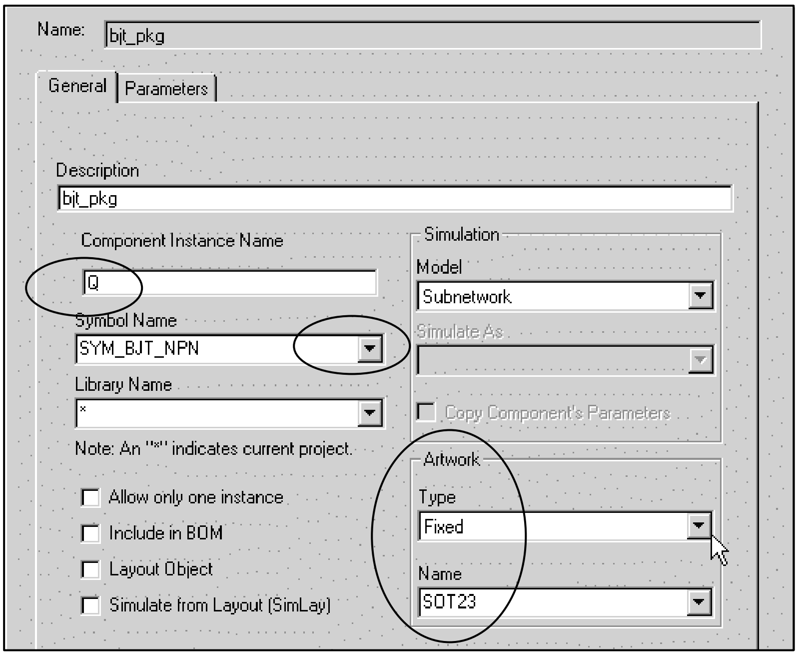


c. Return to the schematic. Click: View > Create/Edit Schematic.

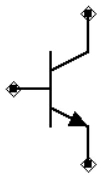
5. Set up the schematic Design Parameters.

a. Click File> Design Parameters and the dialog appears

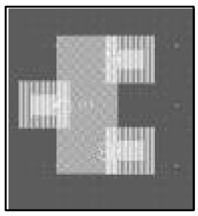
b. In the General tab, make theses changes: 1) change the Component Instance Name to Q, 2) change the Symbol Name to SYM_BJT_NPN by clicking the arrow and selecting it (this is the built-in symbol), 3) in the Artwork field, select Fixed and



Built-in symbol: SYM_BJT_NPN



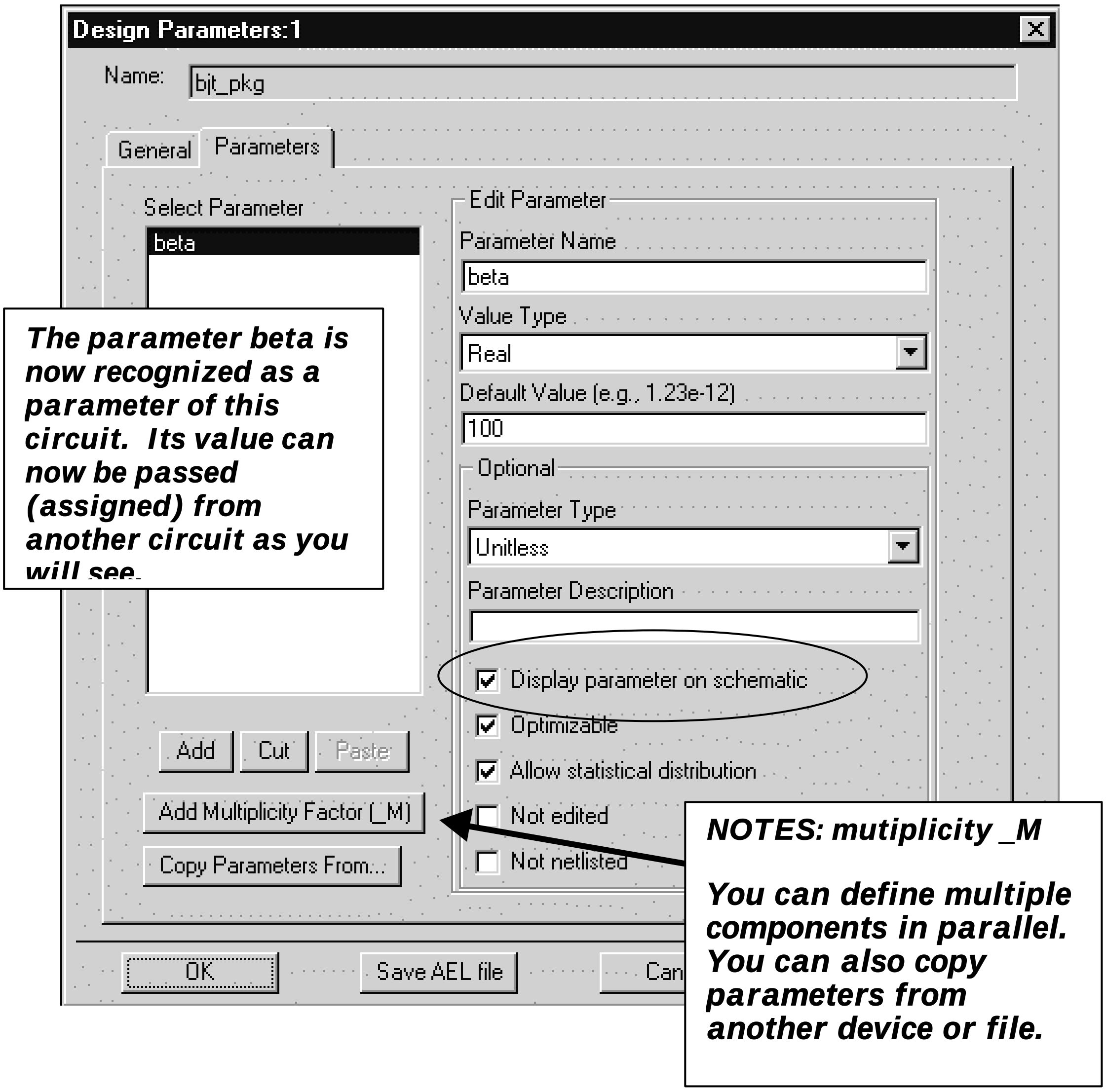
SOT23 fixed artwork



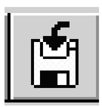
SOT23 as shown here.

c. ***Click Save AEL File to write these changes but do not close this dialog yet because you still need to set other parameters.***

d. ***Go to the Parameters tab. In the Parameter Name area, type in beta and assign a default value of 100 by clicking the Add button. Then check the box to Display the parameter as shown here. Click the OK button to save the new definitions and dismiss the dialog.***



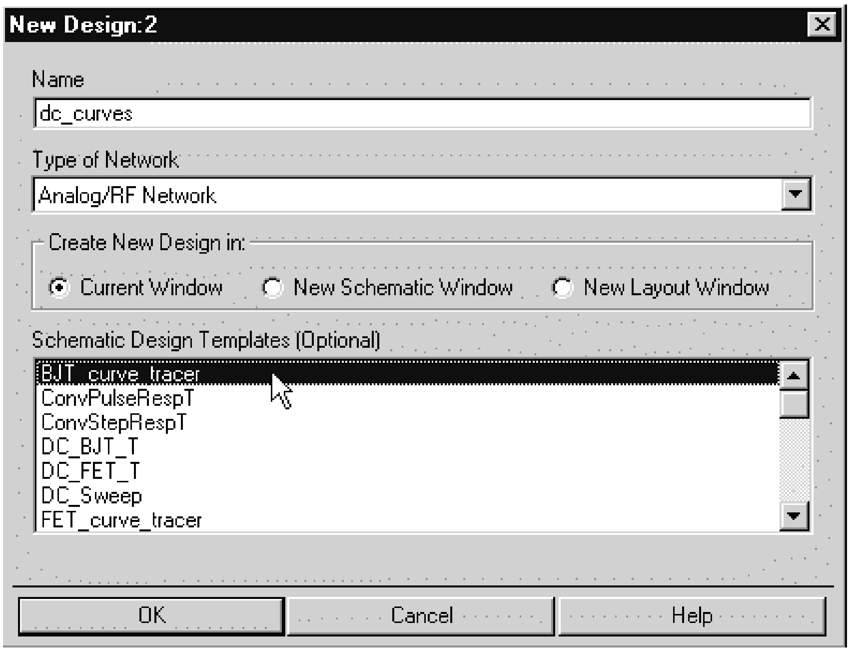
e. ***In the schematic window, Save the design (click the icon shown here) to make sure all your work creating this sub-circuit will not be lost. In the***

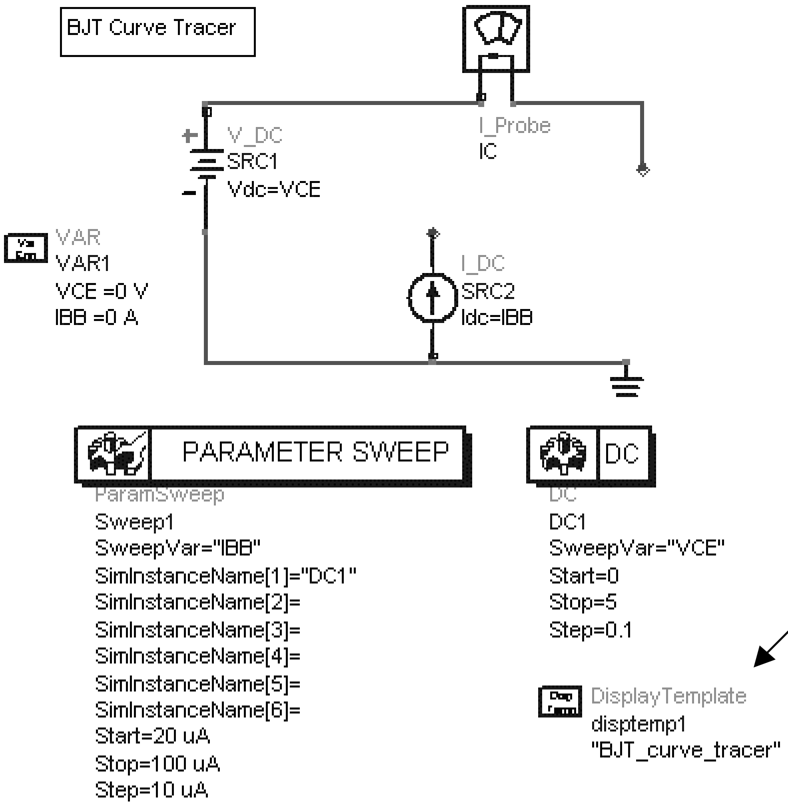


next steps, you will see how the Design Parameters will be used.

6. Use a curve tracer template to test the bjt_pkg subcircuit.

a. In the current schematic of the bjt_pkg, click File > New Design. When the dialog appears, type in the name: dc_curves and select the BJT_curve_tracer template as shown. Click OK and a new schematic will be created with the template, ready to insert bjt_pkg.

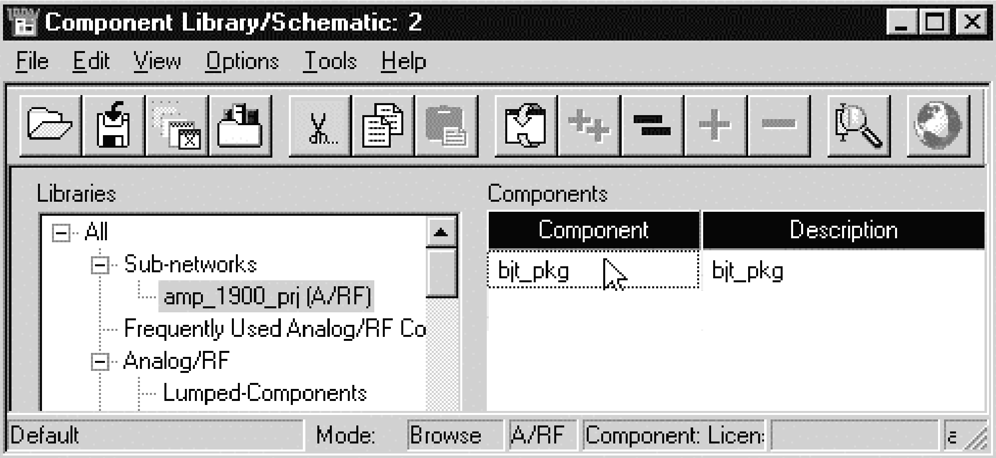




This is a data display template – it will automatically plot the curves.

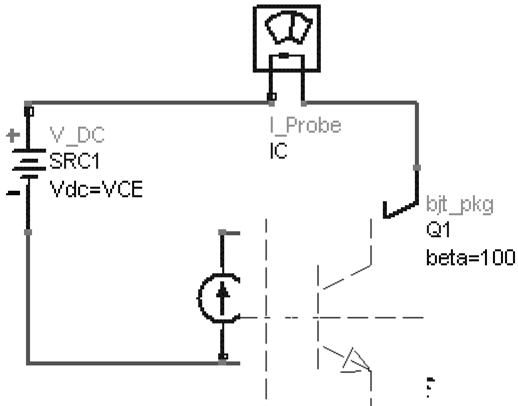
b. ***Save the design and then click the Component Library icon (shown here).***

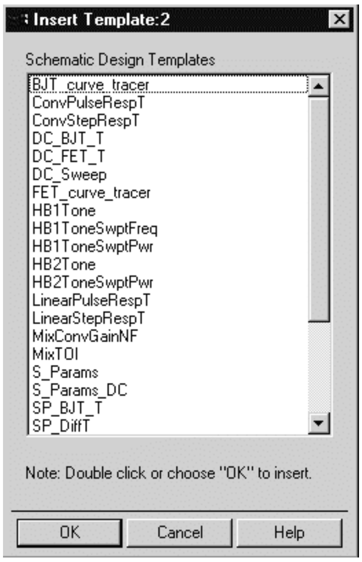
c. ***When the dialog opens, select the amp_1900 project and click on the bjt_pkg sub-circuit and insert it into schematic as shown here. Every circuit that you build will be available in the project library as a sub-***



circuit.

d. ***Connect the bjt_pkg component as shown. You may have to adjust the wires and text (F5) to make it look good. Also, you can now close the library window and save the dc_curves design again - it is good practice to save often.***



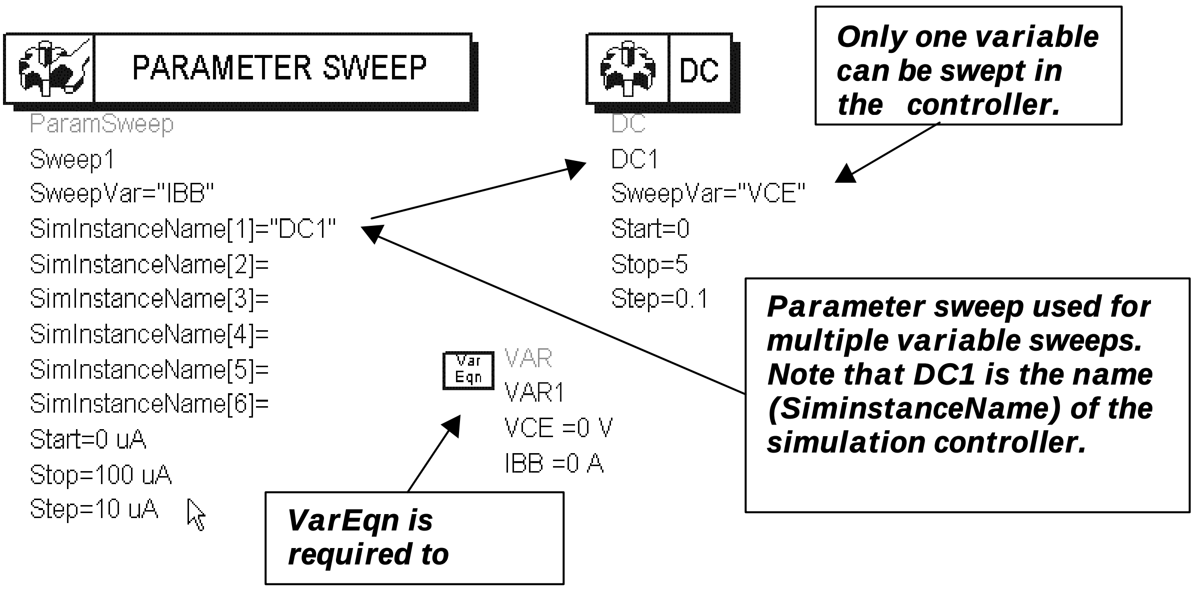


NOTE on templates: You can also insert templates using the schematic window command: Insert > Templates. Many templates have predefinedvalues, node names (wire labels), and variables. Therefore, you may have to make modifications to fit your circuit. Also, many of these templates have data display templates to automatically plot the data. These same data display templates are available in the data display window. In general,

using templates is very efficient and time-saving if you know how to use ADS.

7. Modify the template Parameter Sweep.

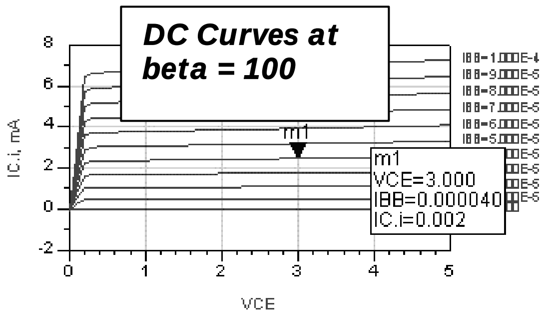
a. Change the Parameter Sweep IBB values to: 0 uA to 100 uA in 10 uA steps as shown here. Do not change the DC simulation controller default settings for sweeping VCE – they are OK. Notice that the VAR1 variables (VCE=0 and IBB=0) do not require modification because they are only required to



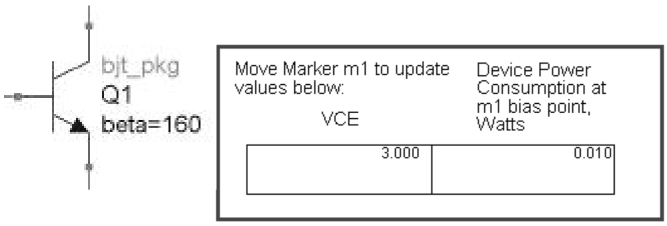
initialize (declare) the variable for the simulator.

8. Simulate at beta=100 and 160.

a. Simulate (F7) with Beta = 100: After the simulation is finished, the data display should automatically appear with the curve tracer results. Try moving the marker and watch the updated values appear.



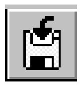
b. Simulate again with Beta = 160: On the schematic, change the value of beta = 160 and simulate again. You should see the updated values:



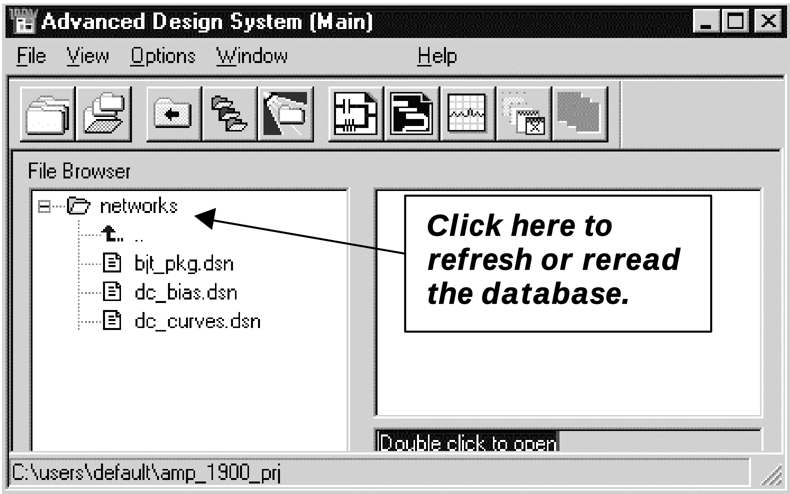
VERIFY for beta = 160 and VCE = 3V: IBB=40 uA and IC=3 mA with about 10mW of consumed power.

9. Open a new design and check all your files in the Main window.

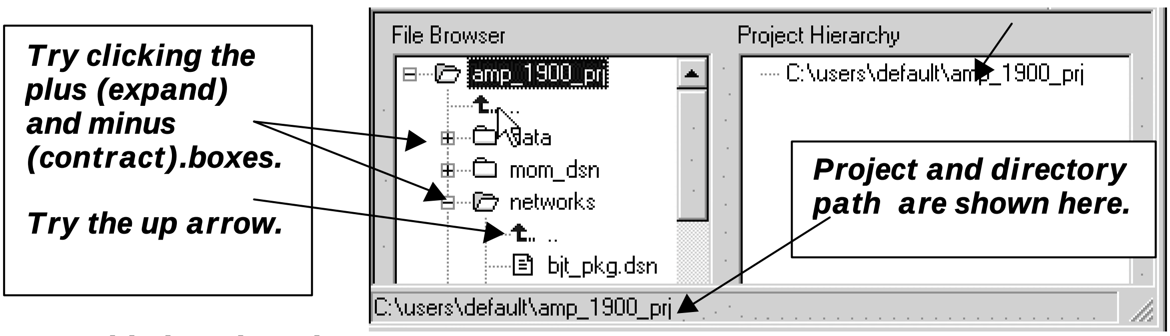
a. Save the current schematic. In the same window, create a new design (without a template) named: dc_bias. Click on the Save Current Design icon (shown here) so that the current design is written into the ADS database.



b. Now, check the ADS Main Window: you should have 3 designs in the networks directory: bjt_pkg, dc_curves, and dc_bias. You may have to click the file browser networks to refresh.

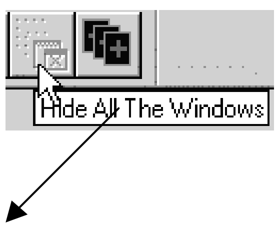


c. In the File Browser area, click on the plus / minus boxes and the up arrow (or two dots). This allows you to see the files you have created in this project. Remember: you can only work in one project at a time, but you can copy files from other projects



and bring them in.

d. Finally, try the Show / Hide all windows feature. This is used for security or to find other open

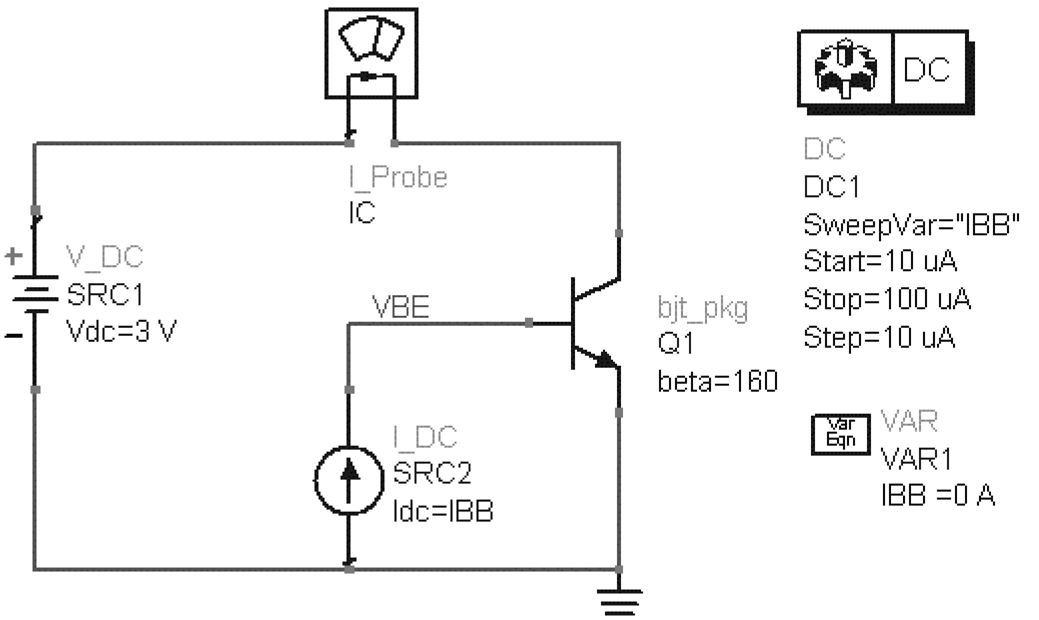


windows that are not ADS. In this case, only the Main window remains.

10. Set up a new schematic and simulate with a parameter sweep.

NOTE on setting up a parameter sweeps: If only one variable is swept, use the Simulation controller (Sweep tab). However, if more than one parameter is swept, a Parameter sweep component is required as in the templates you just used. In general, all simulation controllers allow you to sweep only one parameter or variable.

<u>BUILD the circuit - insert components and then wire it - the steps follow:</u>



a. ***Insert the bjt_pkg using library icon or the component history. Now push into the bjt_pkg and click File > Design Parameters. Reset the beta parameter default to 160, pop out and delete the bjt_pkg and reinsert it – beta is now 160 whenever you use the modeled circuit.***

b. ***From the Probe components palette, insert a current probe and rename it IC instead of I_Probe1 as shown here.***

c. ***From the Sources-Frequency domain palette or using component history (type in the component name), insert V_DC = 3 V and I_DC = IBB as shown here. If you edit these components, you can see they have other parameters which can be useful in some simulations.***

d. ***Wire the components together and add the ground (ground icon).***

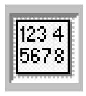
e. ***From the palette, insert a DC simulation controller. Edit (double click) the controller and go to the Sweep tab and assign:***

IBB: 10 uA to 100 uA in 10 uA steps. Then go to the Display tab and check the settings to be displayed as shown. Then click Apply and OK.

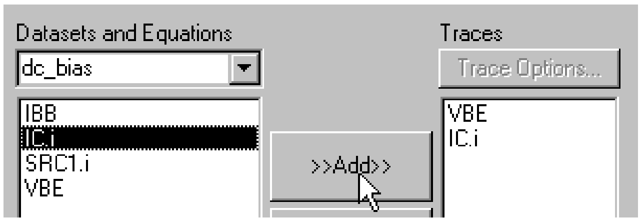
f. Insert a VAR (click icon) variable equation. Use the cursor on the screen to set IBB=0 A. This must be done to initialize (declare) the variable to be swept. Also, if you edit the VAR you will see how the dialog allows you to add more variables or edit existing ones.

g. Insert a wire label VBE at the base. The voltages at that node will appear in the dataset for use in calculating bias resistor values.

h. Simulate and plot the data. When the data display opens, insert a list of VBE and IC.i only. Because you swept IBB to get these values, IBB will automatically be included.



NOTE on results: As you can see, with 3 volts across the device, 40 uA of current results in about 799 mV across VBE with about 3.3 mA of collector current. If you want, draw a box around the values at 40 uA IBB.



IBB	IC.i	VBE
1.000E-5	599.8uA	754.8mV
2.000E-5	1.430mA	777.1mV
3.000E-5	2.349mA	789.9mV
4.000E-5	3.325mA	798.8mV
5.000E-5	4.341mA	805.7mV
6.000E-5	5.389mA	811.3mV

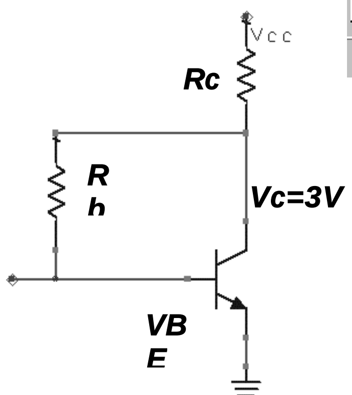
i. Save the design and data display.

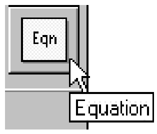
11. Calculate bias values Rb and Rc for a grounded-emitter circuit

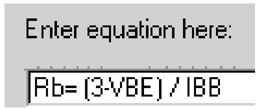
a. In the data display, insert an equation and type: Rb = (3 – VBE) / IBB.

b. Select the Rb Eqn and then use the keyboard Ctrl C and Ctrl V to copy the equation – it will become Rb1.

c. Highlight the Rb1 equation as shown and type in the syntax for Rc = 2 / (IC.i + IBB).





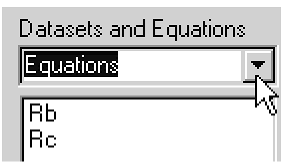


Eqn Rb= (3-VBE) / IBB

Eqn Rb1=(3-VBE) / IBB

Eqn Rc = 2 / (IC.i + IBB)

d. Insert a List, scroll down to the Equations menu (shown here), and add Rb and Rc. Then edit both column headings on the list with a bracketed [3] as shown. This is because the value of 40uA IBB is in the 4rd position in the array row which begins with zero: 0, 1, 2, 3, etc. You can also use Plot Options to add a label to the list:



IBB	Rb[3]	Rc
1.000E-5	224518.376	3279.559
2.000E-5	111142.843	1379.583
3.000E-5	73669.708	840.609
4.000E-5	55029.037	594.350
5.000E-5	43886.113	455.442
6.000E-5	36479.142	367.037

Bias Resistor Calculations

Rb[3]	Rc[3]
55029.037	594.350

12. Set up a biased network.

Now that you have the calculated bias resistor, you can test the bias network with resistors Rb and Rc.

a. ***Save the current design (dc_bias) with a new name: dc_net. By this time, you should know how to do this (File > Save Design As).***

b. ***Delete the current source IBB, the I_Probe, and the Var Eqn.***

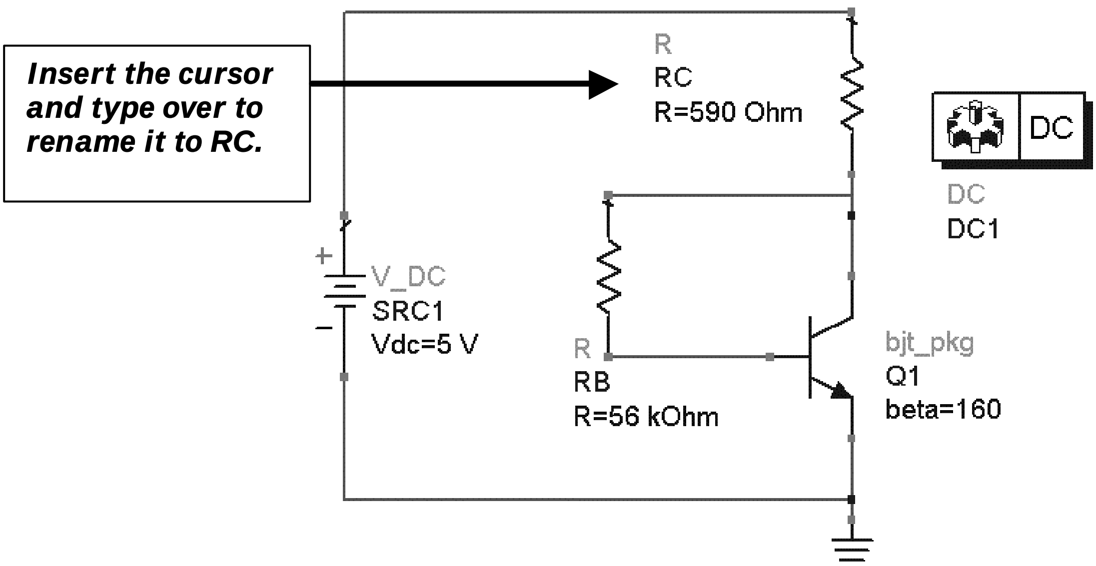
c. ***Go to the Lumped Components palette and insert resistors for the base (56 kOhm) and collector (590 Ohm) as shown. Notice that "R" appears in the history list when you do this. Also, with the resistor attached to your cursor, click the –90 rotate icon to increment it 90 degrees – then insert it.***

d. ***Change the instance R names to RC and RB as shown.***

NOTE on components with artwork: Later on (after the last lab), you can easily and quickly change to lumped components with artwork by changing the component name – for example, change R to R_Pad1, C to C_Pad1, L to L_Pad1, etc. Then you can create a layout of the schematic. For now, use the lumped without artwork.

e. ***Set the V_DC supply: Vdc = 5 V. Wire the circuit and organize it.***

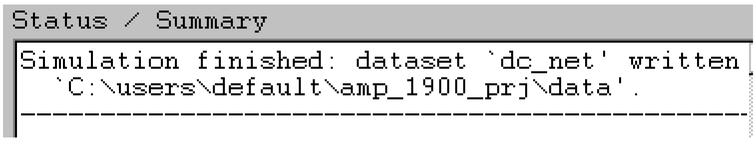
f. ***Delete the DC simulation controller and put a new one in its place – this is faster and more efficient than removing the sweep settings. Because there is no sweep, you do not have to set anything to check DC values.***



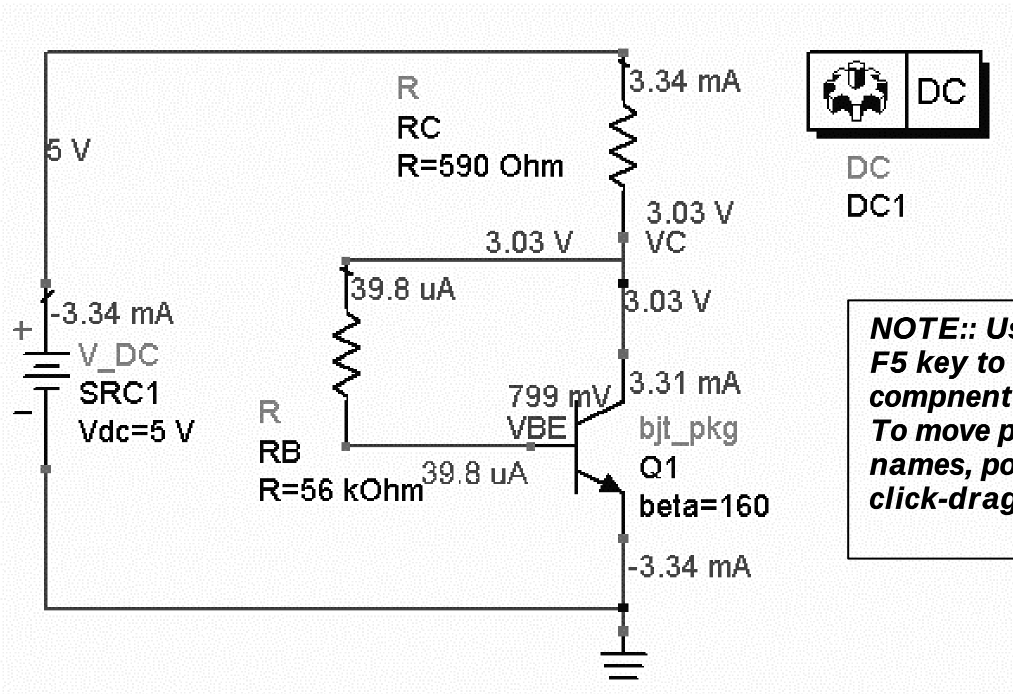
13. Simulate and annotate the DC conditions.

a. Use the Simulate > Simulation Setup (or Hot Key "S" if you have set it) and turn off the opening data display. For this step, you do not need to have the data display opening.

b. Press the F7 keyboard key and the simulation will be launched with the dataset name that is the same as the schematic – this is the default. You can verify this by reading the status window:– simply wait for the simulation to complete.



c. Annotate the current and voltage by clicking on the menu command: Simulate > Annotate DC Solution. If necessary, move components or component text (F5 key) to clearly see the values of voltage and currentt. Be sure that you have the same values shown here. If not, check your work, including the sub-



NOTE:: Use the F5 key to move compnent text. To move pin/node names, point-click-drag,

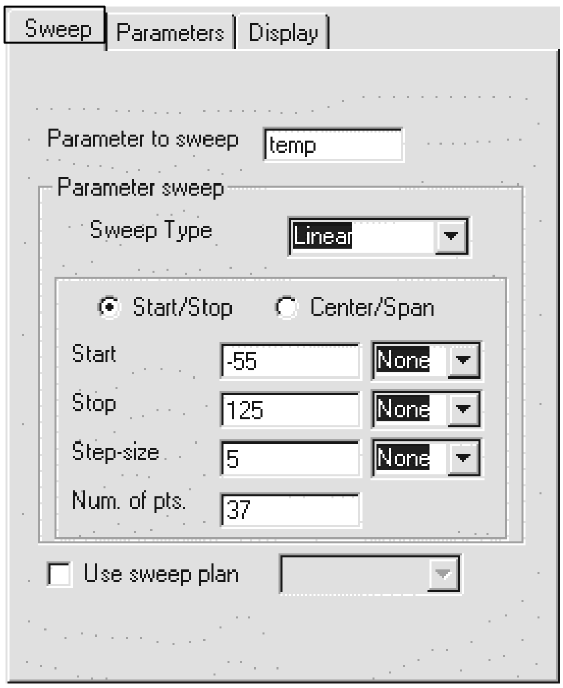
circuit.

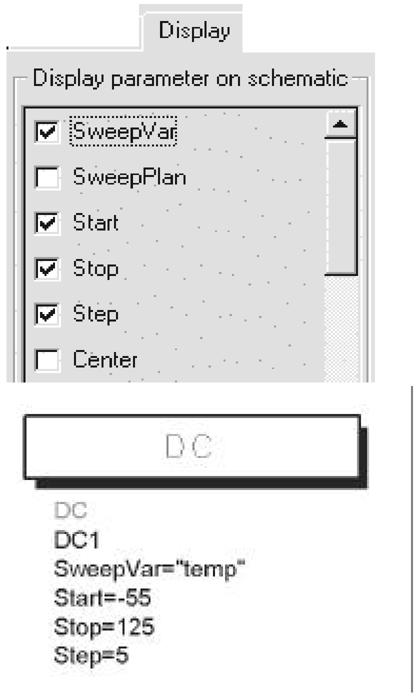
d. ***Clear the annotation, click: Simulate > Clear DC Annotation and then Save all you work.***

14. OPTIONAL: Sweep Temperature

a. ***Edit the DC controller – select it and click the edit icon.***

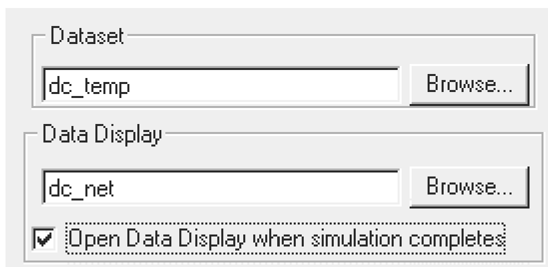
b. ***In the Sweep tab, enter the ADS global variable temp (default is Celsius) as shown here and enter the sweep range: -55 to 125 with step size = 5. Also, in the Display tab, click the boxes to display the annotation on the controller – click Apply to see it and OK to dismiss the dialog.***





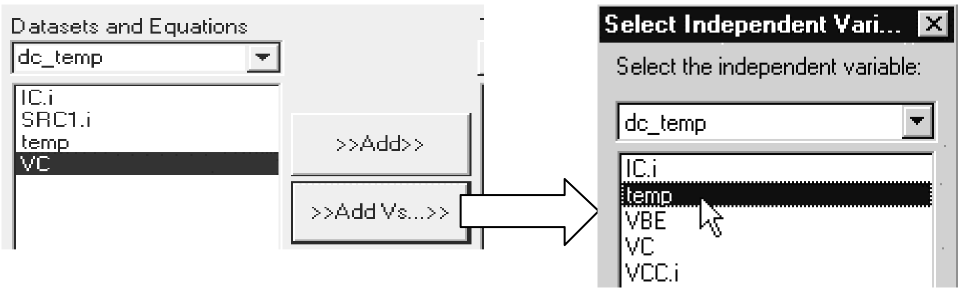
c. ***Insert VC and VBE node labels.***

d. ***Set the simulation dataset name to dc_temp, and check the box to open the***

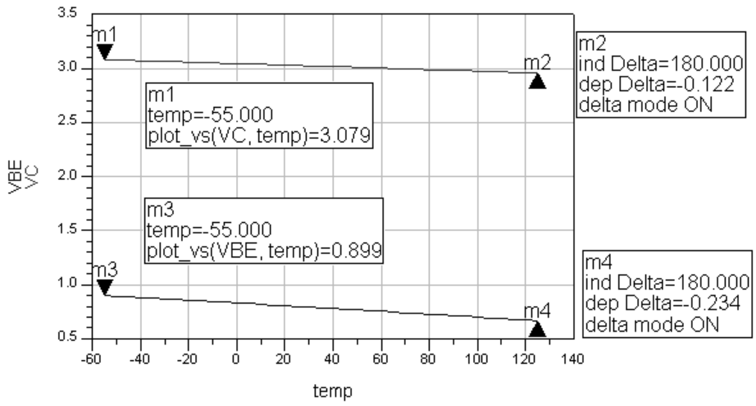


data display dc_net. Click Apply and then Simulate.

e. ***Plot the results in a rectangular plot as VC vs temp and VBE vs temp - you should be able to do this as shown:***



f. ***Put two markers on each trace in delta mode to see the change in voltage as temperature changes. The plot should look like the one shown here: collector voltage decreases at almost one-half the rate of VBE as the temperature increases. You can use this temperature sweep method (sweeping the global variable "temp') for any simulation in the future.***



<u>EXTRA EXERCISES</u>:

1. ***Use the template SP_NWA_T to generate S-parameters for bjt_pkg at all the bias points.***

2. ***Plot current (probe: IC.i) vs. temperature. Or, try setting up a passed parameter for temperature (Temp = 25 in the options controller). The Options controller is in every simulation palette and can be used to set convergence for DC and constant simulation temperature.***

3. Replace the Gummel-Poon model card with another model (Mextram) and resimulate. Afterward, compare the results.

THIS PAGE LEFT INTENTIONALLY BLANK.

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

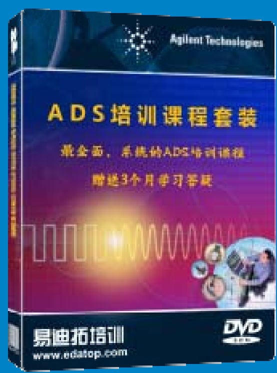
射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

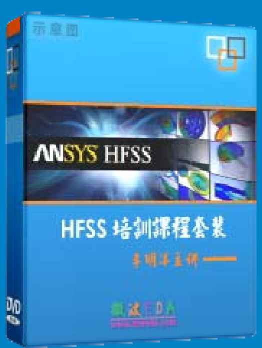
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...



课程网址： http://www.edatop.com/peixun/ads/13.html



HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html